AIAA JOURNAL
Vol. 39, No. 8, August 2001

Film Cooling Injection Hole Geometry: Hole Shape Comparison for Compound Cooling Orientation

Ian Gartshore* and Martha Salcudean†
University of British Columbia, Vancouver, British Columbia V6T 1Z4, Canada
and
Ibrahim Hassan‡
Concordia University, Montreal, Quebec H3G 1M8, Canada

Film cooling, for gas turbine blades or other applications, is done by injecting cool air through a row of holes in a solid wall, past which hot gas is flowing. To investigate the effect of hole shape directly, the present experimental and computational study compares the film cooling effectiveness of two sets of compound-oriented holes, one square and the other round, both placed (alternatively) in a plane wall. Both have the same cross-sectional area, and both are tested in the same facility at the same three blowing ratios R: 0.5, 1.0, and 1.5. Numerical simulations are made using the standard k–ε turbulence model. Film cooling effectiveness is measured using a flame ionization detector. Results show that the holes perform quite differently, the square holes being slightly superior only very close to the injection point and only at low R. For all higher blowing ratios and larger downstream distances investigated, the round holes are better due to the lower integrated momentum flux away from the wall plane at the hole exit. The marked differences between the effectiveness of round and square holes confirms that hole exit geometry is an extremely important factor in film cooling design, even at compound orientation angles.

Nomenclature

A	=	area
C	=	coolant concentration in the freestream
D	=	round hole diameter
d	=	effective cross-sectional hole diameter, equal to jet width in the case of square hole
k	=	turbulence kinetic energy
L	=	length of coolant hole tubes
Q	=	average normal volume flux $\int_A W \cdot \mathrm{d}A$
R	=	jet-to-crossflow velocity (blowing) ratio, V_j/V_∞
Re	=	Reynolds number
V, W	=	mean velocity components in the y and z directions, respectively
V_j	=	bulk jet velocity
W_M	=	average value of W taken over the jet exit area
X, Y, Z	=	axes of tunnel coordinate system
α	=	injection angle (Fig. 1a)
β	=	hole axis orientation angle with respect to the freestream direction (Fig. 1b)
β_z	=	momentum correction factor defined by Eq. (2)
ε	=	turbulence dissipation rate
η	=	adiabatic film cooling effectiveness
$\bar{\eta}$	=	spanwise-averaged film cooling effectiveness
ν	=	kinematic viscosity
ρ	=	density

Subscripts

j	=	jet
∞	=	crossflow

Received 12 February 2000; revision received 14 February 2001; accepted for publication 21 February 2001.

*Professor Emeritus, Department of Mechanical Engineering; gshore@unixg.ubc.ca. Senior Member AIAA.
†Professor Emerita, Department of Mechanical Engineering.
‡Assistant Professor, Department of Mechanical Engineering.

Introduction and Review

FILM cooling is used to protect solid walls from high-temperature gas flowing past the wall. The cooler film of air, spreading out from row(s) of injection holes, insulates the wall from the harsh effects of the external gas. On injection, the coolant begins mixing with the external flow, and the most successful film cooling systems are those that produce the greatest coverage of the wall with the least mixing, using the least possible coolant.

Unfortunately, these are often conflicting requirements suggesting somewhat different designs, so that compromises must be made. Referring to Fig. 1, there are two angles that define the orientation of the simplest film cooling hole: the first angle α defines the hole inclination with respect to the wall (here assumed plane) and the second angle β defines the hole axis orientation with respect to the freestream direction. The hole may, in addition, be flared near its exit, and additional angles and dimensions are needed to define this local distortion of the hole geometry near the injection point. Holes are usually placed in a spanwise row, and this introduces another parameter, the spanwise distance between hole centerlines.

Clearly a low value of α, which produces little immediate separation, will keep mixing to a minimum and will keep the coolant fluid close to the wall. Practical considerations require that this angle be at least 30 or 35 deg, and structural constraints may require that larger angles be used. If β is zero, the case for streamwise injection, the mixing is minimized, but the coverage is also a minimum for a given hole geometry and spacing. Spanwise injection, for which $\beta = 90$ deg, can produce good coverage but also involves high mixing, so that the film cooling provided by these holes may be effective close to injection but less effective farther downstream.

A compromise orientation is the compound injection hole, for which α is the smallest practical value and β is set at 30–60 deg. Although attractive on average across the span, this compromise can leave alleys of the wall well cooled and others relatively unprotected by the coolant, which flows downstream from the injection holes in fairly distinct lines (in steady flow). These coolant footprints gradually spread and, therefore, overlap only some distance away from the injection holes, where the film cooling protection is reduced by mixing of the coolant with the freestream gas. If the effectiveness of a cooling arrangement is judged only by its spanwise average, the compound orientation appears to be quite attractive.

Many experimental studies have been done on film cooling arrangements, with the objectives of understanding this complex flow and of devising the best possible film cooling injection scheme. The

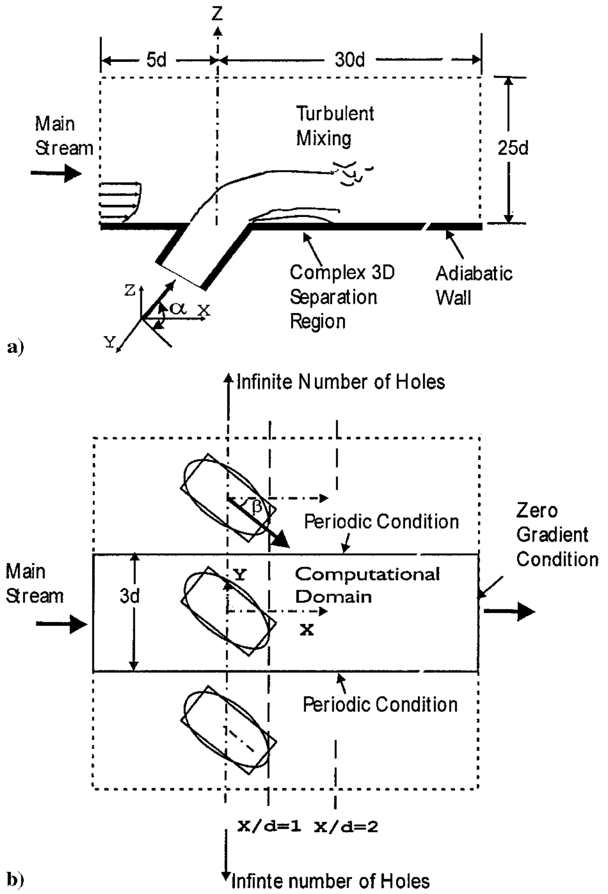

Fig. 1 Jet schematic and computational domain: a) general view and b) plan view of round and square holes superimposed on each other.

effect of the angle β has been studied recently by Findlay et al.[1] for square jets inclined at $\alpha = 30$ deg and by Jung and Lee[2] for round holes at $\alpha = 35$ deg. The measurements and visualizations from these and other research reports show that the flow downstream of the injection holes, arranged in a cross-stream row in which the holes are about three hole diameters apart, is dominated by a single vortex if the angle β is greater than about 15 deg. For smaller angles, the familiar counter-rotating vortex pair (CRVP) emerges, as reported by Andreopoulos and Rodi[3] among others.

The detailed investigation of the effects of hole geometry reported by Haven and Kurosaka[4] linking vorticity from various parts of the injection hole to that observed in the wake, considered only normal injection ($\alpha = 90$ deg), for which the CRVP is clearly present. Detailed conclusions for the compound angled jets are difficult to make on the basis of this study because the wake is very different in the two cases. One general conclusion from Haven and Kurosaka[4] stands out for possible use, however; the hole geometry with the largest projected spanwise length is likely to provide the least separation or liftoff of the injected jet fluid and, therefore, the most effective film cooling close to the hole. The present paper investigates the effects of hole shape for compound cooling holes ($\alpha = 30$ deg and $\beta = 45$ deg).

Other parameters that have been investigated include spanwise hole length,[5] freestream turbulence,[6] hole entrance effects,[7] hole exit tapering,[8,9] and density ratio effects.[10] Tabs and struts have been suggested as devices that reduce separation or liftoff of the injected fluid and, therefore, improve film cooling effectiveness (see, for example, Shih et al.[11] or Zaman and Foss[12]). Only recent references (describing measured or calculated investigations) have been given here; earlier reports may be traced through the reference lists of the papers cited.

From these reports and their references, the following broad generalizations can be made about the geometries most appropriate for film cooling of gas turbine blades:

1) Compound-oriented cooling holes are superior to those with streamwise injection ($\beta = 0$) or normal injection ($\alpha = 90$ deg). Spanwise injection ($\beta = 90$ deg) may be preferable to compound cooling for some applications because the coverage of the coolant on the wall just downstream of injection is more uniform than is the case for values of β close to 45 deg.

2) Low values of R (blowing ratio, V_j/V_∞) or I [momentum ratio $(\rho_j V_j^2)/(\rho_\infty V_\infty^2)$] are preferable for good film cooling effectiveness. Here V represents the mean velocity magnitude and subscripts represent the jet or local freestream conditions. Very low values of R are not safe in practice because of unavoidable fluctuations in conditions near the injection point, so that $R \approx 1.0$ is more common, with corresponding values of I that reflect the different densities of coolant and hot gas.

3) Low values of α are best, but 20–45 deg (depending on hole location) are the smallest angles that are practical.

4) Spanwise hole spacing is very important, and values of z/d of about 3 are common, where d is an effective cross-sectional hole diameter. Smaller values of this ratio are difficult to use in practice, and no optimum value larger than 3 has been determined.

5) Flared holes can provide significant improvements over simple straight hole geometries, particularly at high values of R. A carefully chosen increase in the local injection hole area can slow the coolant just at the hole exit and can direct the coolant so that injection is more nearly tangential to the wall surface. Spanwise flaring and local reductions in the effective value of α are, therefore, advantageous, providing more effective film cooling. This implies that the hole shape, particularly near the hole exit, can strongly affect the film cooling effectiveness of a row of cooling holes, and with today's manufacturing techniques, a wide variety of hole shapes can be considered.

6) Calculations using the usual steady-state engineering turbulence models can give fair predictions at low values of R but consistently fail to predict measured values adequately at higher blowing rates, for all geometries. Grossly unsteady effects, which are apparently present in the actual flow, are not captured by the calculations, and this aspect may be responsible for the inadequacy of the numerical predictions.

7) Reynolds number effects may be important in determining separation characteristics, but no clear guidelines have been developed. Experiments often keep a constant value of V_∞ so that increases in Re necessarily increase the hole Reynolds number (dV_j/ν_j). The effects of these two variables are, therefore, difficult to untangle. The hole Reynolds number is usually about 5×10^3 in practice.

8) Where the local freestream velocity is varying in the streamwise direction (implying associated streamwise pressure gradients), as is the case in curved leading-edge applications, the local (at injection) freestream velocity can be used as a first approximation to apply data gathered from flat plate (zero pressure gradient) experience. More experimental work needs to be done in cases in which a pressure gradient and/or a curvature of the wall exist. This is particularly true because most common numerical predictions are not yet reliable in this complex flow.

9) Flow near the entrance of the injection hole affects the cooling effectiveness significantly because the coolant tube is short (typically $4d$ or less). Flow separation at the entrance to the tube, whether induced by crossflow in the interior or by the sharp corners at the entrance to the tube (or both), can radically alter the flow at the hole exit and, therefore, can affect the film cooling effectiveness over the entire external wall. A changing internal flow, therefore, implies a changing external effectiveness.

10) A high level of turbulence in the freestream flow can increase the degree of mixing between the coolant and the external flow and, therefore, degrade the film cooling effectiveness. Turbulence scales approaching the hole size (or larger) are particularly dangerous in this regard.

11) The density of the coolant may be as much as three times the freestream density. The use of the momentum ratio I goes some way toward accounting for the effects of density, but does not completely correlate all results.

Because film cooling effectiveness is known to be particularly sensitive to hole shape near the hole exit, the present experimental and computational studies were done to investigate the effects of relatively small changes in injection hole shapes, for a row of typical compound-oriented holes. Specifically, round and square cross section holes without flare have been compared.

As reported in this paper, we have found experimentally that there are significant differences between the results from these two hole shapes and that these differences persist for some distance downstream of the injection location. This observation originally raised the possibility that the square holes produced more detailed flow structure than the round holes, which in turn suggested that the round holes could be more accurately simulated by the usual engineering turbulence models (such as k–ε) than would be possible for holes of square shape.

To investigate the effects of hole shape on film cooling effectiveness, the present work addresses the following questions:

1) What differences in the film cooling effectiveness are produced by holes of round and square cross-sectional shape when both have the same compound orientation ($\beta = 45$ deg) and the same inclination ($\alpha = 30$ deg)? Which of these two geometries are more effective and why?

2) Is the effectiveness provided by round holes more accurately simulated numerically by engineering turbulence models than that of square holes?

Experimental Arrangements

The experiments were performed in an open-circuit, low-speed, blower-type wind tunnel. The experimental arrangements were the same as those reported by Findlay et al.[1] and will be described only briefly here. Details may be found in Refs. 1 and 13.

Six jet holes were arranged in a spanwise row across the floor of a small wind-tunnel test section, 405 mm wide and 270 mm high. The holes were either square with cross-sectional dimension $d = 12.7$ mm or round with diameter $D = 14.4$ mm. The cross-sectional areas of these two hole geometries were equal within 0.6%. Both sets were compound oriented such that $\alpha = 30$ deg and $\beta = 45$ deg. For both cases, the spanwise spacing between hole centrelines was $3d$, and the length of the coolant hole tubes, L, was $4d$. A plan view of the exit of the two hole geometries (round and square, superimposed on top of each other) is shown in Fig. 1b. Measurements were taken at five locations downstream of the injections holes: at the downstream edge of the hole ($x/d = 1$ for square holes or 0.857 for round holes) and at $x/d = 2$, 3, 5, and 8.

Coolant injection velocity was held constant in every case so that the Reynolds number (dVj/ν_j) was held constant, with a value in these tests of about 5×10^3. Coolant and freestream air densities were identical. Blowing ratios R of 0.5, 1.0, and 1.5 were investigated for both geometries. Upstream boundary layers were turbulent because they were tripped by a wire placed on the tunnel floor well upstream of the injection holes. Boundary-layer characteristics were essentially constant for all tests with the boundary-layer thickness equal to about $2d$. The shape of the boundary layer also remained essentially unchanged for all blowing ratios (shape factor of about 1.4). Thus, the boundary-layer momentum thickness θ remained essentially constant for all of the blowing ratios. The values of R_θ (Reynolds number based on θ), therefore, varied from about 1.2×10^3 to 4×10^2 as the freestream velocity was varied from 18 to 6 m/s to vary the blowing ratio from 0.5 to 1.5. Freestream turbulence levels were low, ranging from 1% for $R = 0.5$ to 3% for $R = 1.5$.

The mean effective jet penetration and the adiabatic film cooling effectiveness η were measured with a flame ionization detector (FID). For these measurements, a small percentage (less than 0.5%) of propane was added to the coolant air well upstream of injection. The air downstream of injection was sampled through a rake of 11 fine tubes (0.5 mm in outside diameter) spanning $3d$. Samples taken

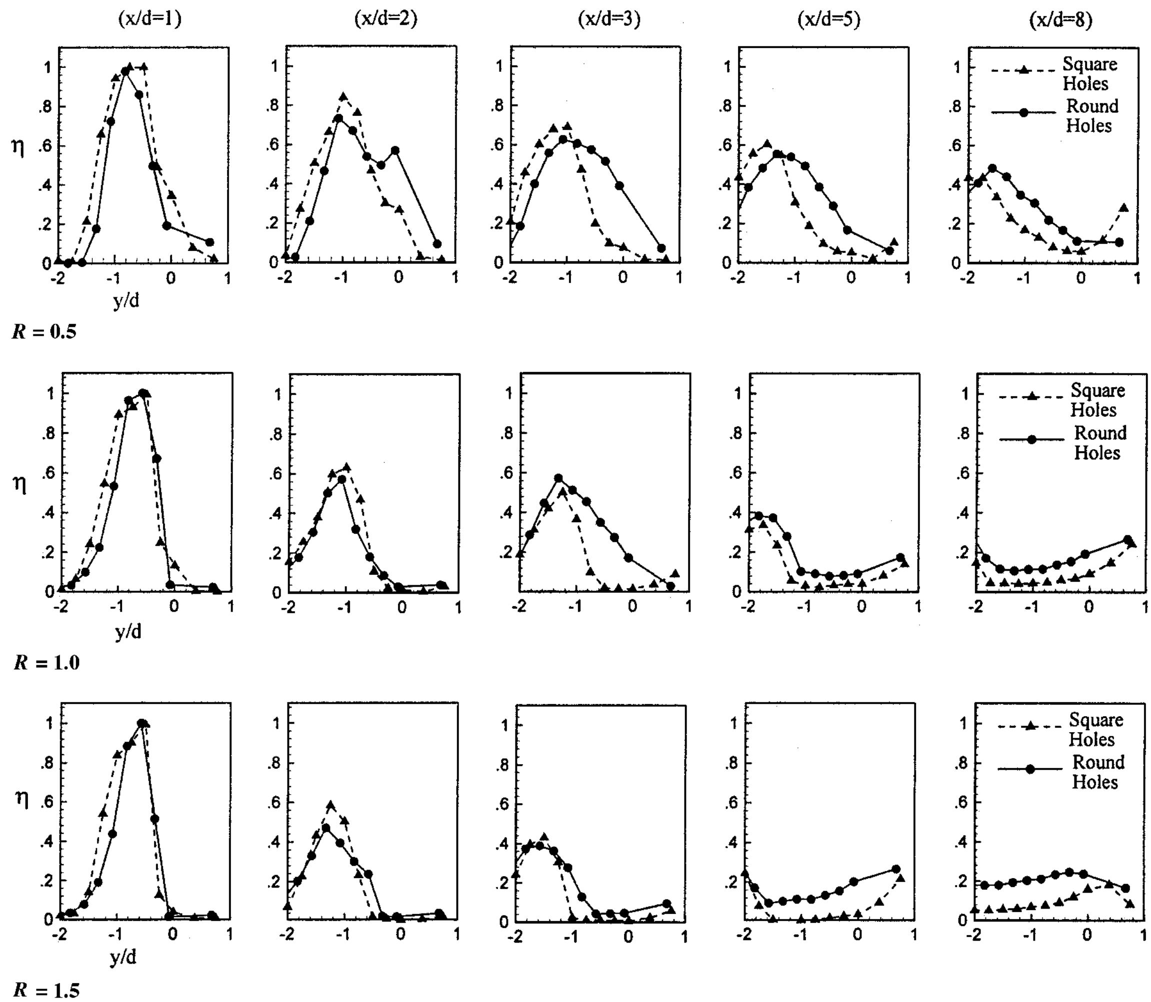

Fig. 2 Measured spanwise distributions of square and round jet effectiveness at different values of R.

on the floor were directly converted to values of η by the mass/heat analogy through the formula

$$\eta = (C - C_\infty)/(C_j - C_\infty) = (C)/(C_j) \qquad (1)$$

because the propane concentration in the freestream, C_∞, was equal to zero. Samples taken at other positions were treated with the same formula, producing relative jet fluid concentrations at each point sampled. More details on the FID technique may be found in Ref. 13.

Uncertainties in the experimental measurements have been estimated following the procedure recommended by Kline and McClintock.[14] The FID measurements of η are accurate to within ± 0.02 where values range from 0 to 1. Measurements in the location of the FID rake are accurate top within ± 1 mm in Z where the rake is above the wall. When the rake is placed directly on the wall, measurements are made at an effective height of 0.025 mm (the radius of the measurement tubes). Because the gradient of η is zero at the wall, the measurements at 0.025 mm are still accurate within ± 0.02 at this height. Values of R (the blowing ratio) rely on measurements of flow rate and freestream velocity. The resulting R values are accurate to $\pm 2.75\%$ within 95% confidence limits. Further discussion of the accuracy of measurements made by the present methods can be found in Refs. 1 and 15.

Measured Results

The measured spanwise distributions of film cooling effectiveness for both round and square holes are shown in Fig. 2. The first downstream location is listed as $x/d = 1$ for both tests although (as already noted) the tests for the round holes were actually made at $x/d = 0.857$ so that they were just downstream of the round injection holes. Although there is little difference between the η distributions for round or square geometries very close to the injection holes, the round holes are clearly superior at larger downstream distances. Broader coverage is evident for round hole injection at $x/d = 3$, 5, and 8, with less mixing at even the largest downstream distance.

The differences are even more clearly displayed by spanwise averages of η, shown in Fig. 3. There, it can be seen that the square holes are more effective, on average, only at the most upstream location and only for the lowest blowing ratio (0.5). This is contrary to the general notion expressed by Haven and Kurosaka,[4] whose results are admittedly only for normal injection orientation, that holes with the largest projected spanwise width will provide the least liftoff and, hence, the greatest film cooling effectiveness. Here the square holes are actually broader, as can be seen from Fig. 1, but they are generally inferior to the round holes. Mixing is apparently greater for the square holes than for the round, and this produces lower values of η for the square geometry injection.

There is an interesting rise in the values of spanwise-averaged η from the round holes for x/d of 3 or less for the lower blowing ratios (0.5 and 1.0). So marked is the rise in η for the round holes at $R = 1.0$ and $x/d = 3$ that the measurements were repeated for this case to confirm the trend. Good repeatability was found.

An effect in the vortex structure downstream of the injection is causing a relative improvement in the film cooling effectiveness for round holes at x/d between 1 and 3. This is clearly true for lower values of R and is marginally true as well even for $R = 1.5$.

To investigate this effect more clearly, measurements of the jet fluid concentration were taken at $x/d = 1$, 3, and 5 above the floor of the tunnel for the lowest blowing ratio $R = 0.5$. The results are shown as contour plots of jet fluid concentration in Fig. 4. From Fig. 4 it is clear that a jet injected from a square holes rises considerably higher than that from a round hole. The single vortex structure (found in the velocity measurements of Findlay et al.[1]) sweeps freestream fluid close to the wall as the jet rises, reducing the effectiveness of the row of square hole with their high injection. These effects are even more pronounced at $R = 1.0$, from the evidence of Fig. 3; fluid from the round holes rapidly spreads out laterally while sweeping little freestream fluid close to the wall. In consequence, the spanwise-averaged η increases abruptly for round holes, whereas values for the square holes drop steadily as x/d increases.

The detailed study of the evolution of square cross section jets done by Quinn[16] found a particularly complex vortex structure in square jets discharged into still air. In such still-air jets, the vortex

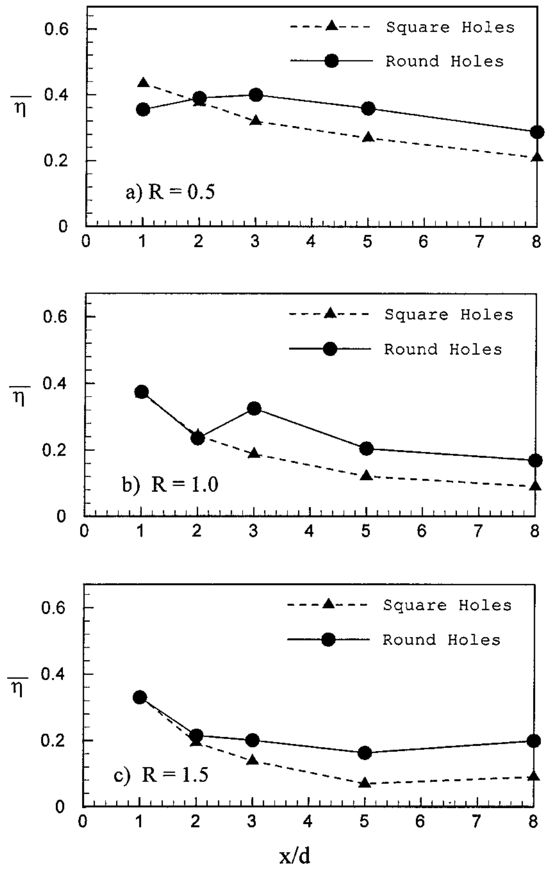

Fig. 3 Measured spanwise-averaged film cooling effectiveness at different values of R.

structure produced by square holes entrains more freestream air, creates greater mixing and more rapid spreading than for comparable round jets. In the present injection into a cross stream, it is not so much the increased mixing but the greater height of the jet fluid away from the floor that produces lower film cooling effectiveness for the square geometries.

Unfortunately, no velocity measurements were taken in the present round hole experiments, so that detailed comparisons with the contour maps of the square jet exit plane reported by Findlay[13] cannot be made. Although both round and square injection holes have the same value of momentum ratio I (for comparable cases) the velocity distribution over the square jet exit is likely to be less uniform than for the round holes (because of dead regions in the corners) so that the actual integrated momentum flux from the square jets will be higher. In other words, the distribution of velocity emerging from the square holes appears to produce a more compact jet with higher peak velocities, greater actual momentum flux, and, therefore, greater penetration than is the case for round jets.

Numerical Methods and Comparisons with Measured Results

The numerical predictions obtainable with standard engineering turbulence models do not provide good predictions of the measured film cooling effectiveness for square jets in crossflow at higher values of blowing ratio R. Zhou et al.,[17] Hassan et al.,[18] and Ferguson et al.[19] showed that there is little improvement in such predictions even when various two-equation turbulence models are tried, and they speculated that there is gross unsteadiness in the flow, which increases mixing very close to the injection hole exit. Ajersch et al.[15] include flow visualization pictures that show unsteady vortex structures for a square normally injected jet at low values of R and low Reynolds numbers. Further support for the presence of unsteadiness is provided by the detailed numerical results of Muldoon and Acharya,[20] who computed the flowfield resulting from the normal injection of fluid from a row of square holes into a cross

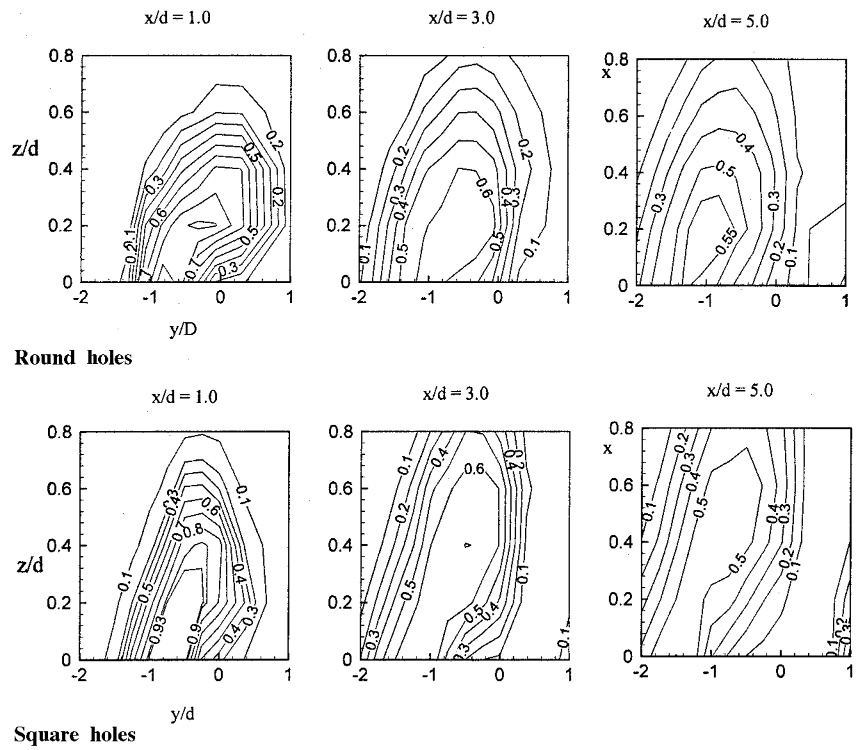

Fig. 4 Concentration (C/Cj) contours in spanwise y–z planes at $R = 0.5$.

stream at $R = 0.5$. Although these authors attempt to make a three-dimensional, unsteady, direct numerical simulation of the entire flowfield, they show clearly that their smallest scales of turbulence are not computed correctly. Numerical dissipation must, therefore, play a significant role in their results. Nevertheless, their numerical predictions show that the flow near the jet exit is grossly oscillatory, with effective Strouhal numbers of about 0.4, based on jet velocity and size. Modest agreement is obtained in comparisons with the detailed measurements of mean quantities reported by Ajersch et al.[15]

Hale et al.[21] evaluated several different turbulence models and wall function treatments available in the commercial computational fluid dynamics code FLUENT, assessing the ability of these combined models to predict the film cooling effectiveness for streamwise blowing from short injection holes (with $L/d = 2.91$, $\beta = 0$ deg, and $\alpha = 35$ deg in the present notation) fed by a narrow plenum. The results of their study indicated that the use of usual wall functions to predict film cooling effectiveness in the near-hole region is problematic due to apparent boundary-layer separation in that region. The use of two-layer zonal models improved the near-hole predictions considerably and gave details of separation just downstream of the holes that were not predicted by the usual engineering wall functions. However, no model of turbulence combined with any of the wall functions that they used was able to give adequate predictions over the entire region downstream of the holes for their blowing ratios of 0.5 and 1.0 .

The small separation region downstream of the blowing holes that Hale et al.[21] identify in their streamwise blowing case will be significantly altered for the present compound angle blowing. Because it is known that the structure within square jets is likely to be more complex than that in round jets and that the separation will be very different for the compound blowing orientation, it was thought that simple engineering models of turbulence might perform better in predictions of compound injection with round holes than with square. This prompted the present numerical calculations of the film cooling effectiveness, testing yet again the efficacy of the two-equation model predictions. The present calculations, like those reported by Hassan et al.[18] use finite volume methods and block structured grids.

The computational domain as shown in Fig. 1 consists of two main blocks or segments. The first block is the jet hole, and the second is the main flow region. The main flow region extends $5d$ upstream of the center of the jet exit, $30d$ downstream, and $25d$ above the plane of the jet exit. The boundary-layer thickness at $x/d = -5$ was set to $2d$ to match experimental observations. The grid used for the computations consists of $34 \times 31 \times 27$ nodes in the main flow block and $11 \times 11 \times 13$ nodes in the jet block. The grid arrangement was based on many preliminary grid-dependence studies. The grid in the jet duct matches with the grids in the main flow region at the interface boundaries. The grid was not uniform in the X, Y, and Z direction with very fine cells near wall.

Boundary conditions were deliberately chosen to be the same in both round and square cases, and the calculations were made by the same curvilinear code.[22] The standard k–ε model with the usual wall functions were used for both geometries. Five types of boundary conditions were used, namely, inlet, outlet, wall, no flux, and periodic. At the upstream edge of the main flow region ($x/d = -5$) the experimental data of the velocity and the turbulent kinetic energy were used to provide the boundary conditions. The turbulence dissipation here was calculated within the boundary layer using a mixing length model. At the duct inlet, uniform distributions of vertical velocity and turbulent kinetic energy were assumed.

The zero-gradient condition was imposed for all dependent variables at the outlet boundary ($x/d = 30$) downstream from the jet center. The bottom wall was assumed to be adiabatic so that zero heat flux was imposed. The same boundary conditions were used for the injection duct walls. The top boundary was located at a distance $25d$ in the Z direction from the jet exit, where an impermeable, free-slip condition was imposed. The standard wall function approach[23] was used near the solid walls of the domain. The periodic (cyclic) condition was used on the boundaries in the spanwise direction, that is, the south and north planes. Such a boundary condition assumes that there are an infinite number of jets in the spanwise direction. Further details regarding the computational method and code are given by He and Salcudean[22] and Hassan et al.[18]

For computational accuracy, the ratio of two adjacent grid sizes in any direction was kept within the range 0.7–1.3. The present grid arrangement was based on many preliminary grid-dependence studies. Whereas the iterations for solution proceed through the conservation of mass, momentum, and energy equations, the convergence criterion was a reduction in the maximum residual of more than four orders of magnitude for each equation. The overall mass balance

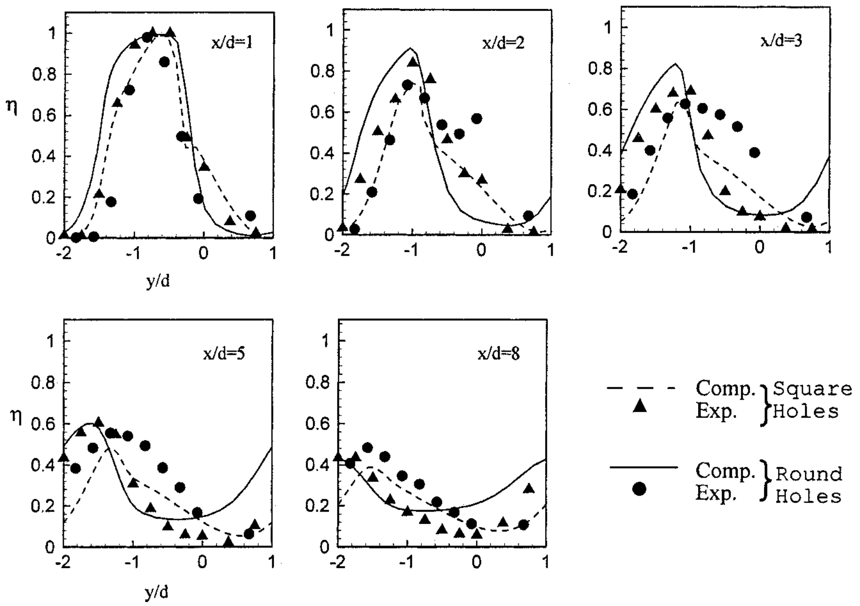

Fig. 5 Comparison of spanwise distribution of square and round jet effectiveness at $R = 0.5$.

between the inlets and the exit of the computational domain was less than 0.5% for every converged solution.

A comparison of the spanwise distribution of film cooling effectiveness for round and square holes, found with both numerical and experimental methods, are shown in Fig. 5 for the blowing ratio $R = 0.5$. Numerical predictions are usually best at this low value of R. Although both calculations agree with experimental values very close to the injection holes, both agree rather poorly at larger x/d.

Some of the differences in performance of the round and square holes can be explained through a consideration of the flux of vertical momentum [in the Z direction (see the axis system in Fig. 1)] in the two cases. For otherwise equal conditions, a larger vertical momentum flux will result in greater liftoff of the injected fluid and generally poorer performance. This effect is most easily evaluated from the calculated numerical results. Similar trends are likely to occur for the measurements as well.

We evaluate the flux of vertical momentum at the exit plane, that is, the integral of W^2 (dA) over each exit plane, nondimensionalizing the result by the average normal volume flux Q (which is the integral of $W \cdot \mathrm{d}A$) and the total hole area A taken in the plane of the wall at the hole exit. Thus, we define

$$\beta_z = \frac{\int_A W^2 \,\mathrm{d}A}{Q \cdot W_M} \tag{2}$$

where $W_M = Q/A$ is the average value of the vertical velocity W taken over the exit area A. The quantity β_z is a particular example of the momentum correction factor found in elementary texts. For a uniform distribution of W, the value of β_z is equal to one. The more non-uniform the flow, the greater the value of β_z becomes. The square hole geometry is likely to produce a less uniform flow at its exit plane than the round geometry and will, therefore, have a higher value of effective vertical momentum flux and a higher value of β_z.

From the computations for $R = 0.5$ (the most accurate computational case), we find that the ratio of β_z for the square hole to that for the round hole is about 1.09. For equal area and volume flux of injected fluid, this means that the square hole, with its lower velocity corner regions, actually delivers a greater integrated flux of vertical momentum than the equivalent round hole and will, therefore, be a less attractive geometry than the equivalent round hole. The measurements reflect this trend.

A concise summary of the experimental and numerical values is provided in Fig. 6, where spanwise-averaged film cooling effectiveness is plotted against x/d for $R = 0.5$ and 1.5. Although one might describe the agreement between computed and measured values as

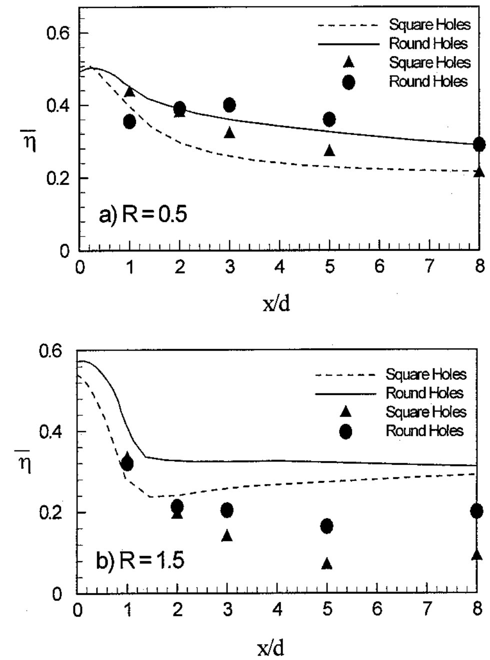

Fig. 6 Comparison of spanwise-averaged film cooling effectiveness at $R = 0.5$ and 1.5: experiments (symbols) and computations (lines).

fair for the lower blowing ratio, neither round nor square results are even remotely predicted at $R = 1.5$. At this high value of R, there is no better agreement for the round hole injection than for the square, dashing any hopes that the former would be easier to predict than the latter. The actual flow is very complex in both these cases, and the present conventional engineering calculations predict effectiveness values that are significantly too high for both geometries. More complicated turbulence models and wall function representations may bring the predictions closer to the observed time-averaged values. It is also possible, however, that grossly unsteady flow is present, adding another dimension of challenge for numerical modelers of such flows.

Conclusions

We can now answer the questions posed at the start of this paper.

1) Are there significant differences in the film cooling effectiveness between holes of round and square cross-sectional shape when

both have the same compound orientation and the same inclination? Which of these two geometries is more effective and why? There is indeed a significant difference between the two geometries, and the round holes are almost always superior, particularly at higher blowing ratios. The difference is because round jets stay closer to the surface than square jets and sweep less freestream fluid beneath them. This in turn is because the square holes deliver a greater flux of vertical (Z direction) momentum than the equivalent round holes, the Z-direction velocity being less uniform for the square holes than for the round at their respective hole exit planes.

2) Is the effectiveness provided by round holes more accurately simulated numerically by engineering turbulence models than that of square holes? There is no great difference between the success of numerical predictions for round holes than for square. Both give fair agreement with measured values at $R = 0.5$, and neither produce reasonable agreement at high values of R.

The preceding conclusions may be generalized to suggest that small changes in hole shape are important for compound-angles injection geometries. It is likely that a round or nearly round hole shape throughout the length of the delivery tube, but particularly near the hole exit, will provide the best film cooling performance. Flared exit shapes have been shown to improve performance for coolant injection, partly because the flared holes can introduce coolant in a direction more nearly tangential to the blade surface. The Z momentum (normal to the surface) cannot be reduced to zero, however, so that, in addition to minimizing this normal momentum component, the best performance will be obtained for flare geometries having the smallest momentum correction factor β_z at the exit plane. Flared region shapes should be designed with both of these criteria in mind.

Common engineering calculation methods (the standard k–ε turbulence model plus usual wall functions) are not adequate to predict values or trends in the measured effectiveness for high values of R for either of these two different injection shapes. Based on the visual evidence reported by Ajersch et al.,[15] the present authors believe that the failure of the usual turbulence models to predict measured values is partly due to the presence of gross unsteadiness in the flowfields, effects that are not modeled correctly by any two-equation or higher-order model. If this is the case, then only unsteady or large eddy simulation calculations are likely to provide adequate predictions of the actual flowfields and, therefore, of the film cooling effectiveness distributions.

Finally, note that the present comparison of the two cooling hole geometries is not complete because heat transfer coefficient data has not been included. As noted by Licu et al.[24] in connection with their η and heat transfer data from square holes, "... regions with highest η do not correspond to regions of lowest heat transfer co-efficient, ..." Heat transfer data are needed for both geometries before the relative merits of round and square holes can be finally assessed. The present data show, however, that the two shapes give quite different effects.

Acknowledgments

Funding for the work on film cooling was provided by Pratt and Whitney Canada and by the Natural Sciences and Engineering Research Council of Canada. Some of the measurements reported here were conducted by R. Bohl, C. Kelly, and A. Kereliuk, and their help is gratefully acknowledged. The assistance received from D. Licu was also much appreciated.

References

[1]Findlay, M. J., Salcudean, M., and Gartshore, I. S., "Jets in Crossflow: Effects of Geometry and Blowing Ratio," *Journal of Fluids Engineering*, Vol. 121, No. 2, 1999, pp. 373–378.

[2]Jung, I. S., and Lee, J. S., "Effects of Orientation Angles on Film Cooling Over a Flat Plate: Boundary Layer Temperature Distributions and Adiabatic Film Cooling Effectiveness," American Society of Mechanical Engineers, ASME Paper 99-GT-143, 1999.

[3]Andreopoulos, J., and Rodi, W., "Experimental Investigation of Jets in a Cross Flow," *Journal of Fluid Mechanics*, Vol. 138, Jan. 1984, pp. 92–127.

[4]Haven, B., and Kurosaka, M., "Kidney and Anti-Kidney Vortices in Crossflow Jets," *Journal of Fluid Mechanics*, Vol. 352, Dec. 1997, pp. 27–64.

[5]Burd, S. W., Kaszeta, R. W., and Simon, T. W., "Measurements in Film Cooling Flows: Hole L/D and Turbulence Intensity," *Journal of Turbomachinery,* Vol. 120, No. 4, 1998, pp. 791–798.

[6]Mehendale, A., and Han, J., "Influence of High Mainstream Turbulence on Leading Edge Film Cooling Heat Transfer," *Journal of Turbomachinery*, Vol. 114, No. 10, 1992, pp. 707–715.

[7]Kohli, A., and Thole, K. A., "A CFD Investigation on the Effects of Entrance Crossflow Direction to Film Cooling Holes," *Proceedings of the 32nd National Heat Transfer Conference*, HTD 350, American Society of Mechanical Engineers, Fairfield, NJ, 1997, pp. 223–232.

[8]Kohli, A., and Bogard, D. G., "Effects of Hole Shape on Film Cooling with Large Angle Injection," American Society of Mechanical Engineers, ASME Paper 99-GT-165, 1999.

[9]Thole, K., Gritsch, M., Schultz, A., and Wittig, S., "Effect of a Crossflow at the Entrance to a Film-Cooling Hole," *Journal of Fluids Engineering*, Vol. 119, No. 3, 1997, pp. 533–540.

[10]Ekkad, S. V., Han, J. C., and Du, H., "Detailed Film Cooling Measurements on a Cylindrical Leading Edge Model: Effects of Free Stream Turbulence and Coolant Density," *Journal of Turbomachinery,* Vol. 120, No. 4, 1998, pp. 799–807.

[11]Shih, T. I.-P., Lin, Y.-L., Chyu, M. K., and Gogineni, S., "Computations of Film Cooling from Holes with Struts," American Society of Mechanical Engineers, ASME Paper 99-GT-282, 1999.

[12]Zaman, K. B. M. Q., and Foss, J. K., "The Effects of Vortex Generators on a Jet in a Crossflow," *Physics of Fluids,* Vol. 9, 1997, pp. 106–114.

[13]Findlay, M. J., "Experimental and Computational Investigation of Inclined Jets in a Crossflow," Ph.D. Dissertation, Dept. of Mechanical Engineering, Univ. of British Columbia, Vancouver, BC, Canada, April 1998.

[14]Kline, S. J., and McClintock, F. A., "Describing Uncertainties in Single-Sample Experiments," *Mechanical Engineering*, Vol. 75, No. 1, 1953, pp. 3–8.

[15]Ajersch, P., Zhou, J.-M., Ketler, S., Salcudean, M., and Gartshore, I., "Multiple Jets in a Cross Flow: Detailed Measurements and Numerical Simulations," *Journal of Turbomachinery*, Vol. 119, No. 2, 1997, pp. 330–342.

[16]Quinn, W., "Streamwise Evolution of a Square Jet Cross Section," *AIAA Journal*, Vol. 30, No. 12, 1992, pp. 2852–2857.

[17]Zhou, J.-M., Salcudean, M., and Gartshore, I., "Prediction of Film Cooling by Discrete-Hole Injection," American Society of Mechanical Engineers, ASME Paper 93-GT-75, 1993.

[18]Hassan, I., Findlay, M., Salcudean, M., and Gartshore, I., "Prediction of Film Cooling with Compound-Angle Injection Using Different Turbulence Models," *Proceeding of the Sixth Annual Conference of the CFD Society of Canada*, CFD Society of Canada, Quebec, Canada, 1998, pp. 419–424.

[19]Ferguson, J. D., Walters, D., and Leylek, J., "Performance of Turbulence Models and Near-Wall Treatments in Discrete Jet Film Cooling Simulations," American Society of Mechanical Engineers, ASME Paper 98-GT-438, 1998.

[20]Muldoon, F., and Acharya, S., "Numerical Investigation of the Dynamical Behavior of a Row of Square Jets in Crossflow Over a Surface," American Society of Mechanical Engineers, ASME Paper 99-GT-127, 1999.

[21]Hale, C., Plesniak, M., and Ramadhyani, S., "Film Cooling Effectiveness Predictions for Short Holes Fed by a Narrow Plenum," American Society of Mechanical Engineers, ASME Paper 99-GT-162, 1999.

[22]He, P., and Salcudean, M., "A Numerical Method for 3D Viscous Incompressible Flows Using Non-Orthogonal Grids," *Journal of Numerical Methods in Fluids*, Vol. 18, No. 5, 1994, pp. 449–469.

[23]Patankar, S., and Spalding, D., "A Calculation Procedure for Heat, Mass and Momentum Transfer in Three-Dimensional Parabolic Flows," *International Journal of Heat and Mass Transfer*, Vol. 15, Oct. 1972, pp. 1787–1806.

[24]Licu, D. N., Findlay, M. J., Gartshore, I. S., and Salcudean, M., "Transient Heat Transfer Measurements Using a Single Wide-Band Liquid Crystal Test," *Journal of Turbomachinery*, Vol. 122, No. 3, 2000, pp. 1–7.

J. P. Gore
Associate Editor